M-Space Slicing Manual

Background Reading

Before submitting prints to the M-Space, read the complete 3D Hubs Knowledge Base available at:
https://www.3dhubs.com/knowledge-base

The Knowledge Base is a great resource and a starting point for 3D printing. If you cannot read the entire Knowledge Base, the following chapters are required reading:

Chapter 1.1 – 1.2
Chapter 2 – All
Chapter 3 .1 – 3.6
Chapter 4.1
Chapter 6.1

M-Space uses Dremel 3D40 IdeaBuilder FDM printers, therefore you should concentrate your reading on FDM topics.

Exporting STL File

When exporting a STL file from either Solidworks or OnShape, ensure that you set the resolution to "Fine". In Solidworks, the setting is located in the Save As dialog under Options > Resolution > Fine. In OnShape it is located under Resolution > Fine in the Export menu. Also, STL files can be exported with dimensional units differing from the modeling environment. Ensure you set the correct STL export unit.

Slicing Software

Models destined to be 3D printed must be "Sliced" into layers by specialized software. The M-Space uses Autodesk Print Studio to perform this task. The software is available on all computers in MC402. If you would like to install a copy on your personal computer, PC and MAC versions are available for download.

To download PrintStuido

1. Go to https://3dprinter.dremel.com
2. Create a profile and sign in
3. Navigate to Support > Downloads > Software
4. Download either the PC or MAC version of Print Studio (NOT older Dremel3D version)

Print Size and Time limitations

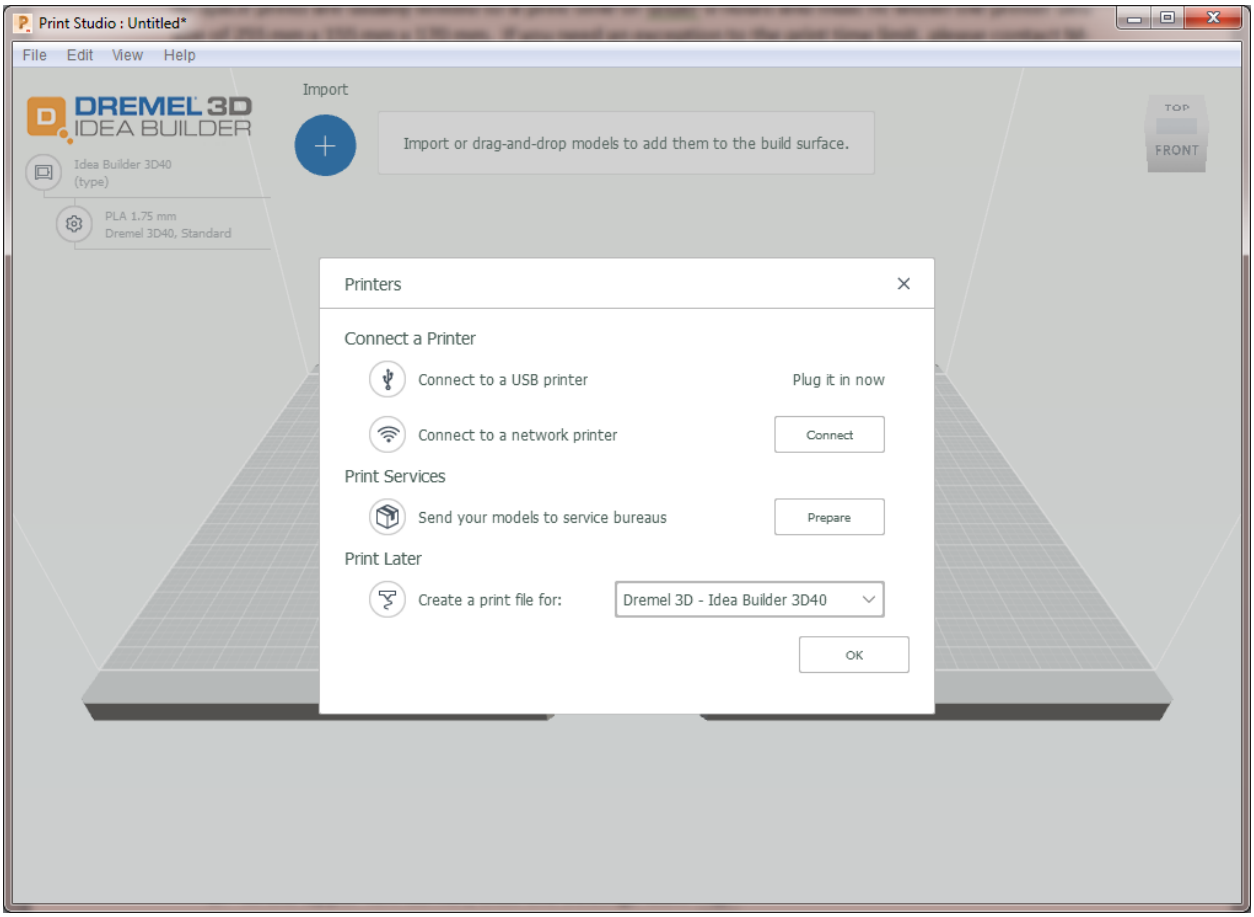
M-Space prints are usually limited to a print time of under 6 hours and must fit within the printer bed size of 255 mm x 155 mm x 170 mm. If you need an exception to the print time limit, please contact M-Space staff.

The following information is taken from the Autodesk PrintStudio online Help Documentation. This is a shortened version including only the necessary steps needed to produce a print in the M-Space. The complete online documentation is available at:

http://help.autodesk.com/view/PRNSTU/ENU/?guid=GUID-B3D68615-05D9-4CEF-8619-9C9959F9861C

Running Print Studio the first time

When running Print Studio for the first time, a “Printers” dialog will appear. Select “Dremel 3D – Idea Builder 3D40” in the “Print Later” section as shown below.



Select a preset print profile

Print profiles contain recommended settings that you can use to get printing quickly. Preset print profiles are created listed for various output quality, such as good, better, or best. The thinner the layer thickness, typically the more detailed the print output. These preset print profiles can't be modified. To do that, create a custom print profile.

Note: Use the “Dremel 3D40 - Standard” profile when preparing prints for the M-Space.

1. In the upper-left corner, click the Settings icon.

2. The Settings dialog box opens.
3. From the Select Print Profile drop-down list, select the desired preset print profile. **(“Dremel 3D40 - Standard”)**
4. Click OK.

To import a model

When you start Print Studio, the first workflow step is Import. Currently, you can import these mesh file formats: 3MF, OBJ, and STL. These file formats don't store information about the model's original unit of measurement (Millimeter, Centimeter, or Inch). So when imported, Print Studio applies its own Units setting to the model. So, for example, even if a model was designed to be 4 inches long and Print Studio's current Unit setting is Millimeter, the model is imported as 4 millimeters long.

Tip: When saving STL files, include the units in the filename to prevent ambiguity.

Import model using one of these methods.

- On the Prep bar, click Import.
- Drag a model into Print Studio.
- Choose File > Import.

Your imported model is selected and centered on the build surface. The Prep bar advances to the Layout workflow step. Here you can arrange and scale imported models.

Important: If the imported model is translucent, that means a portion of the model is outside the build volume (the printable space for the selected printer). Move the model back into the build volume to fix the issue. If the imported model is shaded red that means the model has some errors. Not to worry though. These errors can usually be fixed in the Repair workflow step.

Working with model layout

After importing your model into Print Studio, the Layout workflow step provides you with tools to position each model precisely.

Adjust the view

Once you've imported the model, you may want to change the view or its location in the printable area. You can quickly pan, zoom, and orbit the view as needed.

- To pan the view, press and hold the mouse wheel while dragging the mouse in the desired direction.

- To zoom in, roll the mouse wheel forward.

- To zoom out, roll the mouse wheel backward.

- To orbit the view, right-click while dragging the mouse in the desired direction.

  Tip: You can also use the ViewCube in the upper-right corner of the viewing area to orbit.

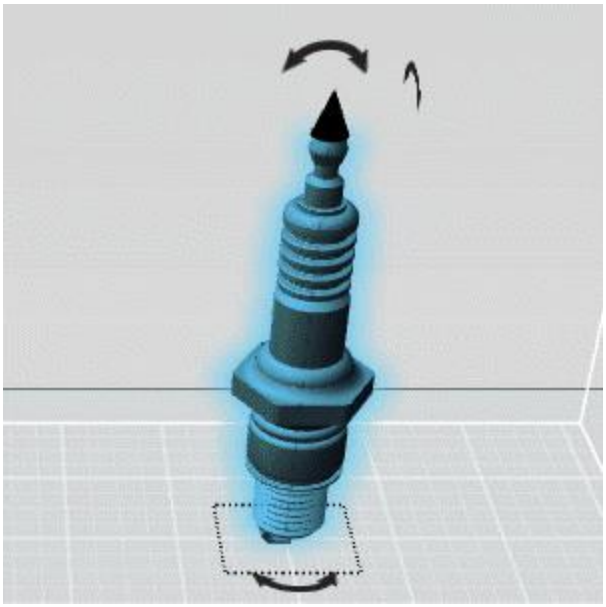
Adjust the position of a model

Besides the ViewCube, you can also accurately position a model.

1. Select the model you want to position.
2. On the vertical toolbar, click the Move icon. 
   Move tools are shown around the model.

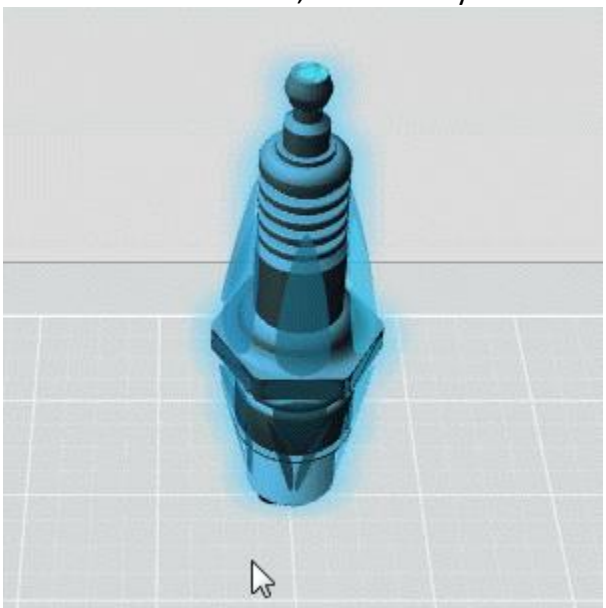
   Tip: You can also use Center, Move to Build Surface, and Auto Layout tools to position the model. These tools are helpful when you have imported more than one model.

3. Use these move tools to rotate, raise, and lower the selected model to the desired position.



Lay a model flat on the build surface

1. Select the model you want to position.
2. On the vertical toolbar, click the Lay Flat icon.

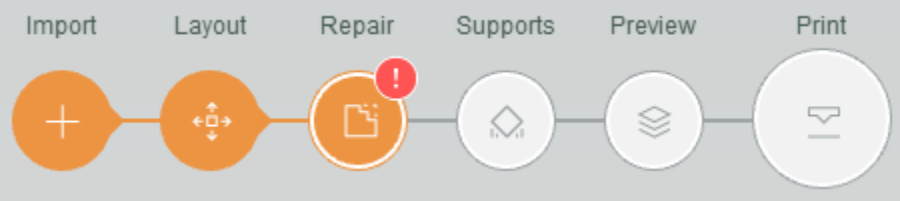


   Faces are selected around the model.
3. Click the selected face you want to lay on build surface.

Working with model repairs

When you click the Repair workflow step, and no model repairs are needed, you can continue directly on to the Supports workflow step. If a model does need repair, Print Studio detects the model issues, such as holes and other geometry defects. You can either repair the model automatically or manually.

On the Prep bar, the Repair icon indicates that issues exist



Automatically repair a model

To automatically repair a model quickly without reviewing the issues first, on the Prep bar, click Supports to advance to the next workflow step. Skipping steps in the workflow causes Print Studio to perform those steps automatically.

If automatic repair fails

If automatic repair fails to fix your model, it is suggested you go back to your modeling software and modify the offending feature. Often adding a radius or fillet in the area fixes any issues.

Working with model supports

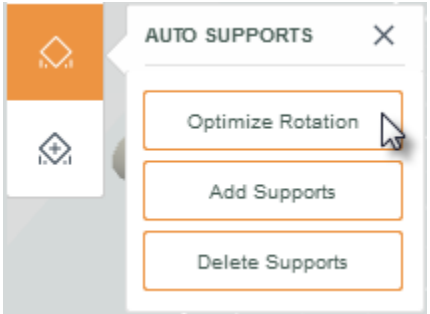
If a model has overhangs or cantilevered sections, adding temporary structural supports before printing can improve the chances that your model prints successfully.

Note: Print Studio is overly cautious with supports and often generates supports where they are not necessary. These supports will increase the time it takes to print. Hence, try to design parts so they do not need support material. Finally, holes in walls generally do not need supports, therefore those can be removed.

Automatically add supports to a model

Depending on how a model is shaped, it may need supports. Print Studio enables you to add them quickly.

1. If necessary, on the Prep bar, click Supports.
2. On the vertical toolbar, click the Auto Supports icon.
3. Optional: On the Auto Supports button menu, click Optimize Rotation.



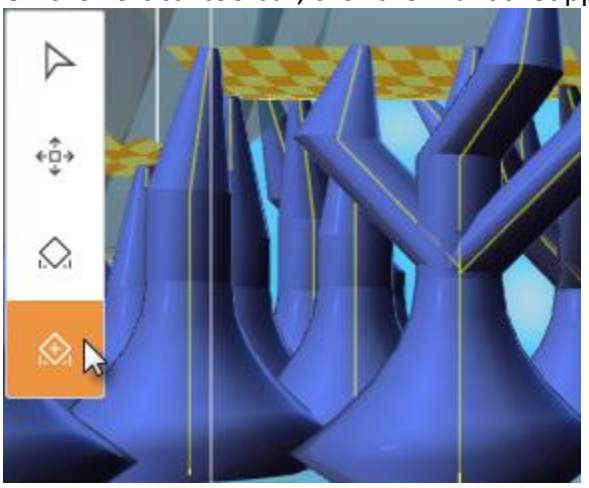
The model is positioned on the build surface to minimize the need for supports.

4. On the Auto Supports button menu, click Add Supports.

Manually add and delete supports

Sometimes, you may want more control over supports, such as inserting extra supports or trimming or deleting existing supports. Print Studio enables you to adjustment supports quickly.

1. If necessary, on the Prep bar, click Supports.
2. On the vertical toolbar, click the Manual Supports icon.



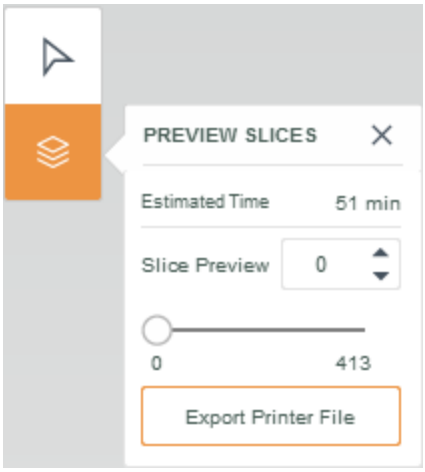
3. Add supports. Areas on a model that need support show a hatched pattern. That pattern is based on the degree of the overhang as indicated in the printer profile's Advanced Settings.
    - To add a support to a model, click a location on the model and then click the build surface. The new support is placed.
    - To add a support to an existing support, click a location on the support and then click the build surface. The new support is placed.
4. Delete supports.
    - To trim a branch from an existing support, click the branch to be trimmed and then click the Delete icon.
    - To delete a support, click the base of the support and then click the Delete icon.

To preview a model

The Preview workflow step in Print Studio slices the model into layers. Use this opportunity to review how your preparations affect the machine instructions before sending them to a printer. These machine instructions determine how and where the printing mechanism moves to produce each layer of material as it prints the model. Reviewing these layers can help you anticipate possible printing issues and give you the chance to correct them before printing.

1. If necessary, on the Prep bar, click Preview.

    On the vertical toolbar, the Preview Slices button menu shows information, such as the estimated print time.



2. To preview the slices, in the Slice Preview box, enter the number of a specific slice. You can also use the slider to go back and forth between the slices.

Save a model

To print a model at the M-Space, you will need to submit a 3PS file. Print Studio can save everything in your scene to a 3PS file, including repairs, supports, and all items outside the build volume. Including models and changes, the 3PS file stores the print profile used when the scene was saved. When you're ready to submit your print to the M-Space, follow these steps to save a 3PS file.

1. Choose File > Save As...
2. Enter a File Name.
3. Click Save.

Reference: http://help.autodesk.com/view/PRNSTU/ENU/?guid=GUID-B3D68615-05D9-4CEF-8619-9C9959F9861C